SSNP162 – Joints 2D and 3D for models `JOINT_MECA_RUPT` and `JOINT_MECA_FROT`

Summarized:

This test has as an aim the validation of the hydro-mechanical elements of joints for the constitutive laws of stoppings `JOINT_MECA_RUPT` and `JOINT_MECA_FROT`.

One tests at the same time pure mechanical modelizations `PLAN_JOINT` in $2D$ (mesh `QUAD4, QUAD8`) and `3D_JOINT` in $3D$ (mesh `HEXA8, HEXA20` or `PENTA6, PENTA15`) with the two models in the presence of imposed pressure. Then, always for same the quadratic elements coupled hydro-mechanical modelizations `PLAN_JOINT_HYME` and `C3D_JOINT_HYME`, with the profile of uplift obtained in computation.

The elements on which these modelizations are pressed are the elements of standard cohesive joints.

Modelization A : `PLAN_JOINT` nets `QUAD4` with `JOINT_MECA_RUPT`
Modelization B : `3D_JOINT` nets `HEXA8` with `JOINT_MECA_RUPT`
Modelization C : `3D_JOINT` nets `PENTA6` with `JOINT_MECA_RUPT`

Modelization D : `PLAN_JOINT` nets `QUAD4` with `JOINT_MECA_FROT`
Modelization E : `3D_JOINT` nets `HEXA8` with `JOINT_MECA_FROT`
Modelization F : `3D_JOINT` nets `PENTA6` with `JOINT_MECA_FROT`

Modelization G : `PLAN_JOINT_HYME` nets `QUAD8` with `JOINT_MECA_RUPT`
Modelization H : `3D_JOINT_HYME` nets `HEXA20` with `JOINT_MECA_RUPT`
Modelization I : `3D_JOINT_HYME` nets `PENTA15` with `JOINT_MECA_RUPT`

Modelization J : `PLAN_JOINT_HYME` nets `QUAD8` with `JOINT_MECA_FROT`
Modelization K : `3D_JOINT_HYME` nets `HEXA20` with `JOINT_MECA_FROT`
Modelization L : `3D_JOINT_HYME` nets `PENTA15` with `JOINT_MECA_FROT`

Modelization M : `PLAN_JOINT` nets `QUAD8` with `JOINT_MECA_RUPT`
Modelization N : `3D_JOINT` nets `HEXA20` with `JOINT_MECA_RUPT`
Modelization O : `3D_JOINT` nets `PENTA15` with `JOINT_MECA_RUPT`

Modelization P : `PLAN_JOINT` nets `QUAD8` with `JOINT_MECA_FROT`
Modelization Q : `3D_JOINT` nets `HEXA20` with `JOINT_MECA_FROT`

Modelization R : `3D_JOINT` nets `PENTA15` with `JOINT_MECA_FROT`

1 Problem of reference

1.1 Geometry

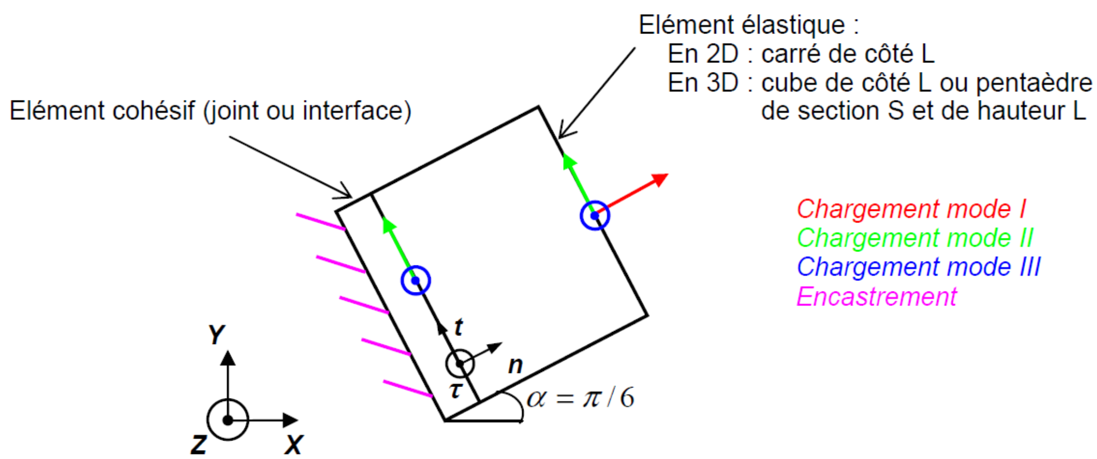

Figure 1 : Representation of the system of two elements in the plane (X, Y) .

One chooses $L=1\,mm$.

1.2 Properties of the material

1.2.1 Model `JOINT_MECA_RUPT`: cubic `material`

JOINT_MECA_RUPT : elastic
Modulus Young: $E=3\times 10^{12}\,Pa$
Poisson's ratio: $\nu=0$

normal stiffness:	$K_N=10^{12}\,Pa/m$	(key word: `K_N`)
tangencial stiffness:	$K_T=2\times 10^{12}\,Pa/m$	(key word: `K_T`)
tensile strength:	$\sigma_{max}=0.1\,MPa$	(key word: `SIGM_MAX`)
tangential regularization of damage	$\alpha=1.5$	(key word: `ALPHA`)
parameter of brittle lissage of fracture	$P_{rup}=0.5$	(key word: `PENA_RUPTURE`)
penalization of the contact	$P_{cont}=3$	(key word: `PENA_CONTACT`)
Density of the fluid (only for modelizations HM)	$\rho_{fluide}=1000\,kg/m^3$	(key word: `RHO_FLUIDE`)
Dynamic viscosity of the fluid (only for modelizations HM)	$\mu_{fluide}=0.001\,Pa.s$	(key word: `VISC_FLUIDE`)
Opening of regularization of flow (only for modelizations HM)	$\delta_{min}=1.E-10\,m$	key word: `OUV_MIN`)

NB: The data materials do not have of course authority to represent a material in particular. They are only intended for numerical tests of validation.

1.2.2 Model `JOINT_MECA_FROT`: cubic `material`

JOINT_MECA_FROT *:* elastic
Modulus Young: $E=3\times10^{12}\,Pa$
Poisson's ratio: $\nu=0$

normal stiffness:	$K_N=10^{12}\,Pa/m$	(key word: `K_N`)
tangencial stiffness:	$K_T=2\times10^{12}\,Pa/m$	(key word: `K_T`)
coefficient of kinetic friction:	$\mu=0.5$	(key word: `MU`)
adhesion (friction with normal loading no one)	$c=10^5\,Pa$	(key word: `ADHESION`)
regularization of the tangent slope in sliding	$\lambda=10^{-6}\,K_T$	(key word: `PENA_TANG`)

NB: The data materials do not have of course authority to represent a material in particular. They are only intended for numerical tests of validation.

1.3 Boundary conditions and loadings

Fixed support : Imposed displacements are null on the face of the cohesive element opposed to the elastic element.
The joint is tilted with 30 degrees compared to the plane horizontal, which gives the following directions of loading according to the mode of request:

In mode I : An imposed displacement U is applied to the face of the elastic element opposed to the joint (see figure 1).

$DX=2.16506351$ $DY=1.250$ $DZ=0$

In mode II : Displacement imposed U is applied on all the nodes of the voluminal element.

$DX=-1.250$ $DY=2.165063509461$ $DZ=0$

In mode III : Displacement imposed U is applied on all the nodes of the voluminal element.

$DX=0.0$ $DY=0.0$ $DZ=2.5$

In mode I the joint follows a standard loading return ticket formulates: initially to the partial opening ($\delta_n>0$), then one passes in compression until $\delta_n<0$ and finally it is discharged up to its point from equilibrium $\delta_n=0$.
In order to supplement the test in mixed mode one partially gives the responsibility in mode I and then one makes a return ticket in mode II (or III according to the modelization).

2 Reference solution

2.1 general Case

In this part, one details the analytical solution in pure I mode in its form $3D$. For plane $2D$ computations, the solution is identical. The component of the jump and the following vector forced τ do not intervene, and it is enough to replace surface S by the length L in the solution.

For the loadings in mode of shears, the elastic element does not play a part.

Initially, the joint follows a loading of type return ticket in mode I what makes it possible to test the profile of the curve forces displacement at various times for this standard loading. In the second part, one charges in a sequential way in mode I, then in mode II (or III according to the modelization). Then one modifies the loading in mode I and discharges in mode II. One thus requests the coupling between the two load patterns (it is checked, for example, that the tangential slope evolves according to the normal loading). One tests also the profile of the curve forces displacement for the tangential stress.

2.2 In pure I mode

One presents the analytical solution of the total response of the system written in the local coordinate system $(\boldsymbol{n},\boldsymbol{t},\boldsymbol{\tau})$. One applies a loading colinéaire to the norm: $\boldsymbol{U}=U\boldsymbol{n}$, the cohesive element opens in pure I mode shear stresses and as well as the tangential jumps remain null. One thus brings back oneself to a scalar problem. One notes $\sigma=\boldsymbol{n}.\boldsymbol{\sigma}.\boldsymbol{n}$ the single non-zero component of the tensor of the stress of the elastic element in the local coordinate system:

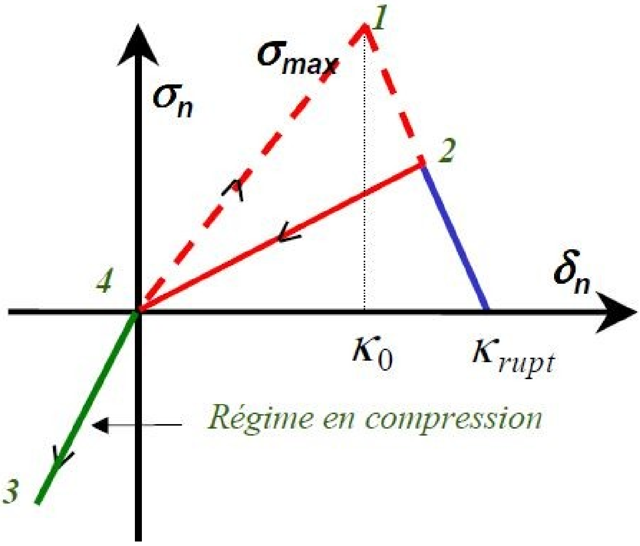

Illustration 1: Loading return ticket in mode I model of JOINT_MECA_RUPT

- **JOINT_MECA_RUPT**

One presents the solution of the total response of the system joint+cube (see document [R7.01.25]).

The equality of the stresses gives:

$$\begin{cases}\sigma=K_N\,\delta_n\\ \sigma=E\,\varepsilon_{cube}=E(U-\delta_n)/L_{cube}\end{cases}$$ in the elastic zone

$$\begin{cases}\sigma=\sigma_{max}-K_N/P_{rup}(\delta_n-\sigma_{max}/K_N)\\ \sigma=E\,\varepsilon_{cube}=E(U-\delta_n)/L_{cube}\end{cases}$$ in the zone of softening

Where $L_{cube}=1$ is the length of the edge of the cube in our case. One notes the maximum opening of the joint until the complete fracture by $U_{max}=\sigma_{max}(1+P_{rup})/K_N$ and the maximum elastic opening par. $U_{el}=\sigma_{max}(K_N L_{cube}+E)/K_N E$

the solution is given by the linear function per pieces:

$\sigma=-K_N E/(K_N L_{cube}+E/P_{cont})U$	si $U<0$	in the compression zone
$\sigma=K_N E/(K_N L_{cube}+E)U$	si $U<U_{el}$	in the elastic zone in tension
$\sigma=K_N E/(-K_N L_{cube}+E*P_{rup})(U_{max}-U)$	si $U<U_{max}$	in the zone of softening
$\sigma=0$	si $U>U_{max}$	complete damage

the force tested is thus given by $F=\sigma S$ where S corresponds to the surface to which one applies the loading.
For the hydro-mechanical modelizations (modelizations G, H, I) the value of the force is given by: $F=(\sigma-p_{fluide})S$ where p_{fluide} the pressure of the fluid contributing to the opening of the joint indicates.

- **`JOINT_MECA_FROT`**

One presents the solution of the total response of the more cubic joined system (see Doc. [`R7.01.25`]).

The equality of the stresses gives:

$$\begin{cases}\sigma=K_N\,\delta_n\\ \sigma=E\,\varepsilon_{cube}=E(U-\delta_n)/L_{cube}\end{cases}$$ in the elastic zone

$$\begin{cases}\sigma=c/\mu\\ \sigma=E\,\varepsilon_{cube}=E(U-\delta_n)/L_{cube}\end{cases}$$ in the plastic zone

Where $L_{cube}=1$ is the length of the edge of the cube in our case. As the maximum stress in mode I for the model of Mohr-Coulomb is given by $\sigma_{max}=c/\mu$, the maximum elastic opening (elastic threshold of tension) is equal to $U_{el}=\sigma_{max}(K_N L_{cube}+E)/K_N E$. One advances up to this value of displacement one checks the value of $\sigma=\sigma_{max}=c/\mu$, then one imposes double displacement is one checks that the value of stress does not evolve. Then in the field of compression the solution is always elastic $\sigma=K_N\,\delta_n$.
The force tested is given by $F=\sigma S$ where S corresponds to surface to which one applies the loading.

For the hydro-mechanical modelizations (modelizations J, K, L) the value of the force is given by: $F=(\sigma-p_{fluide})S$ where p_{fluide} indicates the pressure of the fluid contributing to the opening of the joint.

2.3 In mode II and III pure

the system written in the local coordinate system $(\boldsymbol{n},\boldsymbol{t},\boldsymbol{\tau})$. One applies a loading perpendicular to the norm directly to the joint: $\boldsymbol{U}=U_t\boldsymbol{\tau}$. The cohesive element opens in mode II . If the loading norm does not evolve one brings back oneself to a scalar problem. One notes $\sigma_n=\boldsymbol{n}.\boldsymbol{\sigma}.\boldsymbol{n}$, $\sigma_\tau=\boldsymbol{\tau}.\boldsymbol{\sigma}.\boldsymbol{n}$ the stresses in the local coordinate system.
The loading in mode I being maintained, one charges in mode II (or III according to the modelization). The coupling between the two modes is thus requested. The solution varies according to constitutive law:

•**JOINT_MECA_RUPT**

When the joint is partially open ($\delta_n>0$) the tangential slope evolves according to this normal opening like: $K_T^{evol}=K_T\times(1-\delta_n/L_{CT})$ (see Doc. [R7.01.25])
where $L_{CT}=\sigma_{max}(1+P_{rup})/K_N\tan(\alpha\pi/4)$ is the critical length of tangential damage. The solution is given by: $\sigma_\tau=K_T^{evol}U_t$

•**JOINT_MECA_FROT**

One tests curved force-displacement for the joint, whose normal loading evolves. The joint is partially closed ($\delta_n>0$). The solution is the simple sliding of Mohr-Coulomb. $\sigma_\tau=-\mu\sigma_n+c$ curve is tested at various times (see illustration):

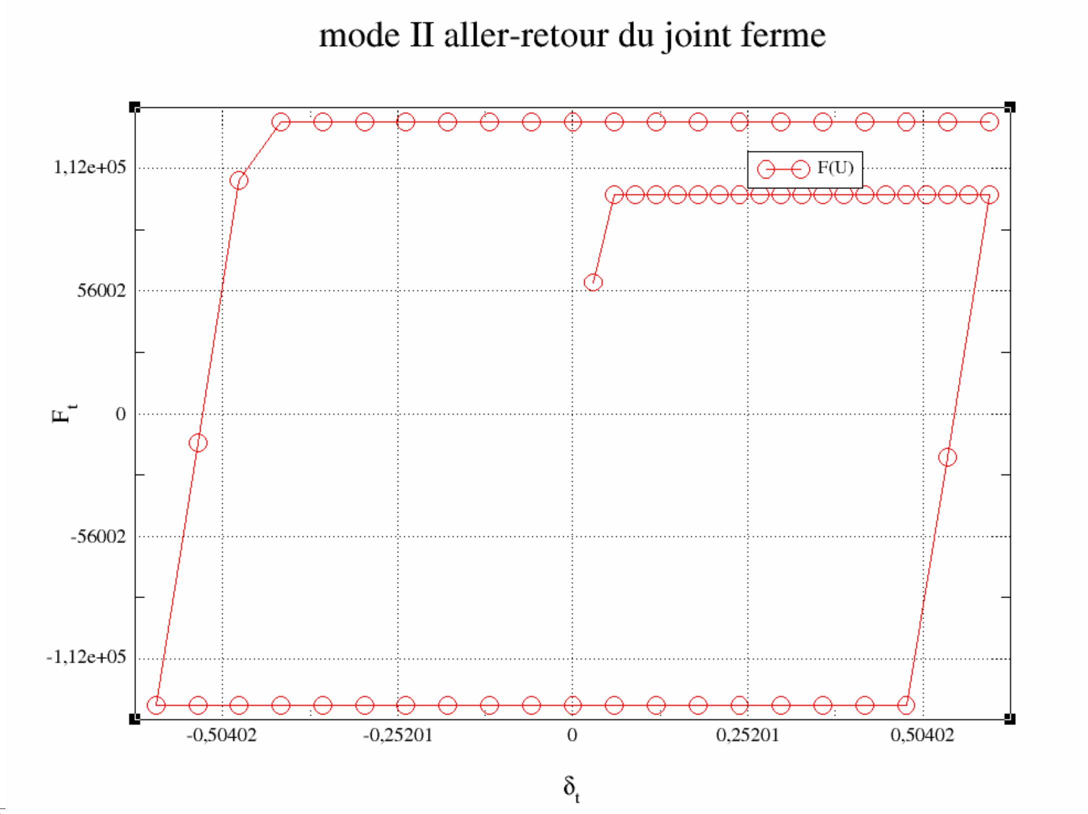

Illustration 2: Curve A forces (tangent) displacement for `model`

3 JOINT_MECA_FROT Modelization

Validation of the joint $2D$ with model JOINT_MECA_RUPT

3.1 Characteristics of the modelization

Modelization in plane strains D_PLAN for the elastic element.
Modelization plane for the element of joint (key word PLAN_JOINT).

3.2 Characteristics of the mesh

Many nodes: 6
the elastic element is a QUAD4.
The element of joint is a QUAD4 degenerated (confused nodes).

3.3 Quantities tested and Mode

results I

One opens the joint until the damage to partiel1voir[1]. $U=\beta\times U_{el}+(1-\beta)\times U_{max}$ Where $\beta=0.2$. Then it passes in the compression zone $U=-U_{el}$. Finally the joint is put in tension until the complete damage $U=U_{max}$. One tests the total response (the resultant of the nodal force, FN) of the system (joined and cubic) in the local coordinate system. The values of reference are analytical (see page 6).

Mode II

the joint is open in mode I until the partial damage, then it is requested in mode II. One tests the tangential slopes of stiffness to two values of normal opening. The tangential character incremental of the evolution makes modify the stress tangential during the partial discharge in mode I what returns complex its estimate analytical. Us it testont thus after the discharge by regression.

One notes by:

$$\sigma_{pena}=\sigma_{max}P_{cont}\times(E+L_{cube}\times K_N)/(E+L_{cube}\times K_N\times P_{cont})$$

$$\sigma_t(\delta_n)=K_T\times(1-\delta_n/L_{CT})U_t$$

Quantity tested	FN	Reference Tolerance % (
	) I	
mode, value with the peak	σ_{max}, either $1.D05$	0.10
FN, damaged value	$\beta\sigma_{max}$, or $2.D04$	0.10
FN, value in compression	σ_{pena}, or $-2.D05$	0.10
	mode II	
FT, for two values of δ_n	$\sigma_t(\delta_n)$	0.10

1 the analytical solution for the notations

4 Modelization B

Validation of the joint $3D$ with model `JOINT_MECA_RUPT`

4.1 Characteristics of the modelization

Modelization `3D` for the elastic element.
Modelization `3D_JOINT` for the element of joint

4.2 Characteristics of the mesh

Many nodes: 12
the elastic element is `a HEXA8`.
The element of joint is `a degenerated` HEXA8 (confused nodes).

4.3 Quantities tested and Modes

results I and II

Identical to modelization A.

Mode III

the joint is open in mode I until the partial damage, then it is requested in mode III . One tests the tangential slopes of stiffness to two values of normal opening.

5 Modelization C

Validation of the joint 3D with model JOINT_MECA_RUPT

5.1 Characteristics of the modelization

Modelization 3D for the elastic element.
Modelization 3D_JOINT for the element of joint

5.2 Characteristics of the mesh

Many nodes: 9
the elastic element is a PENTA6.
The element of joint is a degenerated PENTA6 (confused nodes).

5.3 Quantities tested and Modes

results I and II
Identical to modelization A. It is noted just that $FN=SIGN/2$, because the surface of contact is worth $1/2$.

Mode III
the joint is open in mode I until the partial damage, then it is requested in mode III . One tests the tangential slopes of stiffness to two values of normal opening.

6 Modelization D

Validation of the joint $2D$ with model JOINT_MECA_FROT

6.1 Characteristics of the modelization

Modelization in plane strains D_PLAN for the elastic element.
Modelization plane for the element of joint (key word PLAN_JOINT).

6.2 Characteristics of the mesh

Many nodes: 6
the elastic element is a QUAD4.
The element of joint is a QUAD4 degenerated (confused nodes).

6.3 Quantities tested and Mode

results I

One opens the joint until his threshold of strength to traction2voir it[2]. $U=U_{el}$ Then one draws from advantage to arrive at $U=2U_{el}$. It passes in the compression zone $U=-U_{el}/3$. Finally the joint is reloaded up to its point of equilibrium $U=0$. One tests the total response (the resultant of the nodal force, FN) of the system (joined and cubic) in the local coordinate system. The values of reference are obtained analytically (see page 6).

Mode II

the joint is requested in sliding in mode II . One test E then the value of adhesion c . Then one modifies the normal opening in mode I and one makes the sliding return ticket in mode II for $U_t=\pm U_{el}/3$.

Quantity tested	Reference	Tolerance ($\%$)
	mode I	
FN, value with the peak	σ_{max} either 2.D05	0.10
FN, not variation of the value to the peak	σ_{max} or 2.D05	0.10
FN, value in compression	$\sigma_{max}/3$ or -6.66666667D04	0.10
FN, value at the point of equilibrium	0.0	0.10
	mode II	
FT, value of adhesion	C or 1.D05	0.10
FT, value in sliding and adhesion	$-K_N U_t\ \mu - c$ or -1.2445666D+06	0.10
FT, value in sliding and adhesion	$K_N U_t\ \mu - c$ or 1.244647D+06	0.10

2 the analytical solution for the notations

7 Modelization E

Validation of the joint $3D$ with model JOINT_MECA_FROT

7.1 Characteristics of the modelization

Modelization 3D for the elastic element.
Modelization 3D_JOINT for the element of joint

7.2 Characteristics of the mesh

Many nodes: 12
the elastic element is a HEXA8.
The element of joint is a degenerated HEXA8 (confused nodes).

7.3 Quantities tested and mixed

Mode results I II **,** III
One makes only tests of non regression, by mixing the load patterns. One applies the same loading as in modelization D, but one adds there the loading in the third direction.

8 Modelization F

Validation of the joint $3D$ with model JOINT_MECA_FROT

8.1 Characteristics of the modelization

Modelization 3D for the elastic element.
Modelization 3D_JOINT for the element of joint

8.2 Characteristics of the mesh

Many nodes: 9
the elastic element is a PENTA6.
The element of joint is a degenerated PENTA6 (confused nodes).

8.3 Quantities tested and mixed

Mode results I II , III

One makes only tests of non regression, by mixing the load patterns. One applies the same loading as in modelization D, but one adds there the loading in the third direction.

9 Modelization G

Validation of the joint $2D$ HM with model JOINT_MECA_RUPT

9.1 Characteristics of the modelization

Modelization in plane strains D_PLAN for the elastic element.
Modelization plane for the element of joint (key word PLAN_JOINT_HYME).

9.2 Characteristics of the mesh

Many nodes: 13 (whose 2 nodes with degrees of freedom of pressure)
the elastic element is a QUAD8.
The element of joint is a degenerated QUAD8 (confused nodes).

9.3 Quantities tested and results

to test the hydraulic part, one imposes a pressure $p_{fluide}=500\,Pa$ on the node $NS5$ and one tests the same value on the node $NS7$.

One tests the mechanics in pure I mode. One opens the joint until the damage to partiel3voir[3]. $U=\beta\times\%U_{el}+(1-\beta)\times\%U_{max}$ where $\beta=0.2$. Then it passes in the compression zone $U=-U_{el}$. Finally the joint is put in tension until the complete damage $U=U_{max}$. One tests the total response (the resultant of the nodal force, FN) of the system (joined and cubic) in the local coordinate system. The values of reference are obtained analytically by taking account of the fluid pressure (see page 6).

3 the analytical solution for the notations

10 Modelization H

Validation of joint $3D$ HM HEXA20 with model JOINT_MECA_RUPT

10.1 Characteristics of the modelization

Modelization in 3D for the elastic element.
Modelization 3D_JOINT_HYME for the element of joint

10.2 Characteristics of the mesh

Many nodes: 32 (whose 4 nodes with degrees of freedom of pressure)
the elastic element is a HEXA20.
The element of joint is a HEXA20 degenerated (confused nodes).

10.3 Quantities tested and results

to test the hydraulic part, one imposes a pressure $p_{fluide}=500\,Pa$ on the nodes $NS19$ and $NS20$ one tests the same value on the nodes $NS17$ and $NS18$.

One tests the mechanics in pure I mode. One opens the joint until the damage to partiel4voir[4]. $U=\beta\times U_{el}+(1-\beta)\times U_{max}$ where $\beta=0.2$. Then it passes in the compression zone $U=-U_{el}$. Finally the joint is put in tension until the complete damage $U=U_{max}$. One tests the total response (the resultant of the nodal force, FN) of the system (joined and cubic) in the local coordinate system. The values of reference are obtained analytically by taking account of the fluid pressure (see page 6).

4 the analytical solution for the notations

11 Modelization I

Validation of joint $3D$ HM PENTA15 with model JOINT_MECA_RUPT

11.1 Characteristics of the modelization

Modelization in 3D for the elastic element.
Modelization 3D_JOINT_HYME for the element of joint

11.2 Characteristics of the mesh

Many nodes: 24 (whose 3 nodes with degrees of freedom of pressure)
the elastic element is a PENTA15.
The element of joint is a degenerated PENTA15 (confused nodes).

11.3 Quantities tested and results

to test the hydraulic part, one imposes a pressure $p_{fluide}=500\,Pa$ on the node $NS13$ and one tests the same value on the nodes $NS14$ and $NS15$.

One tests the mechanics in pure I mode. One opens the joint until the damage to partiel5voir[5]. $U=\beta\times U_{el}+(1-\beta)\times U_{max}$ where $\beta=0.2$. Then it passes in the compression zone $U=-U_{el}$. Finally the joint is put in tension until the complete damage $U=U_{max}$. One tests the total response (the resultant of the nodal force, FN) of the system (joined and cubic) in the local coordinate system. The values of reference are obtained analytically by taking account of the fluid pressure (see page 6).

5 the analytical solution for the notations

12 Modelization J

Validation of the joint 2D HM with model JOINT_MECA_FROT

12.1 Characteristics of the modelization

Modelization in plane strains D_PLAN for the elastic element.
Modelization plane for the element of joint (key word PLAN_JOINT_HYME).

12.2 Characteristics of the mesh

Many nodes: 13 (whose 2 nodes with degrees of freedom of pressure)
the elastic element is a QUAD8.
The element of joint is a degenerated QUAD8 (confused nodes).

12.3 Quantities tested and results

to test the hydraulic part, one imposes a pressure $p_{fluide}=500\,Pa$ on the node $NS5$ and one tests the same value on the node $NS7$.

Mode I

One opens the joint until his threshold of strength to traction6voir it[6]. $U=U_{el}$ Then one draws from advantage to arrive at $U=2U_{el}$. It passes in the compression zone $U=-U_{el}/3$. Finally the joint is reloaded up to its point of equilibrium $U=0$. One tests the total response (the resultant of the nodal force, FN) of the system (joined and cubic) in the local coordinate system. The values of reference are obtained analytically by taking account of the fluid pressure what makes shifted the normal curve to the bottom (see page 6).

Mode II

the joint is requested in sliding mode II . One test then the value of adhesion c . Then one modifies the normal opening in mode I and one makes the sliding return ticket in mode II for $U_t=\pm U_{el}/3$. The values of reference are obtained analytically by taking account of the fluid pressure what does not influence the tangential curve (see page 6).

6 the analytical solution for the notations

13 Modelization K

Validation of the joint 3D HM HEXA20 with model JOINT_MECA_FROT

13.1 Characteristics of the modelization

Modelization in 3D for the elastic element.
Modelization 3D_JOINT_HYME for the element of joint

13.2 Characteristics of the mesh

Many nodes: 32 (whose 4 nodes with degrees of freedom of pressure)
the elastic element is a HEXA20.
The element of joint is a HEXA20 degenerated (confused nodes).

13.3 Quantities tested and results

to test the hydraulic part, one imposes a pressure $p_{fluide}=500\,Pa$ on the nodes $NS19$ and $NS20$ and one tests the same value on the nodes $NS17$ and $NS18$.

One makes only tests of non regression, by mixing the load patterns. One applies the same loading as in modelization E, but one adds there the loading in the third direction.

14 Modelization L

Validation of the joint $3D$ HM PENTA15 with model JOINT_MECA_FROT

14.1 Characteristics of the modelization

Modelization in 3D for the elastic element.
Modelization 3D_JOINT_HYME for the element of joint

14.2 Characteristics of the mesh

Many nodes: 24 (whose 3 nodes with degrees of freedom of pressure)
the elastic element is a PENTA15.
The element of joint is a degenerated PENTA15 (confused nodes).

14.3 Quantities tested and results

to test the hydraulic part, one imposes a pressure $p_{fluide}=500\,Pa$ on the node $NS13$ and one tests the same value on the nodes $NS14$ and $NS15$.

One makes only tests of non regression, by mixing the load patterns. One applies the same loading as in modelization E, but one adds there the loading in the third direction.

15 Modelization M

Validation of the joint $2D$ into quadratic with model JOINT_MECA_RUPT

15.1 Characteristics of the modelization

Modelization in plane strains D_PLAN for the elastic element.
Modelization plane for the element of joint (key word PLAN_JOINT).

15.2 Characteristics of the mesh

Many nodes: The 13
elastic element is a QUAD8.
The element of joint is a degenerated QUAD8 (confused nodes).

15.3 Quantities tested and Mode

results I

One opens the joint until the damage to partiel7voir[7]. $U=\beta\times U_{el}+(1-\beta)\times U_{max}$ Where $\beta=0.2$. Then it passes in the compression zone $U=-U_{el}$. Finally the joint is put in tension until the complete damage $U=U_{max}$. One tests the total response (the resultant of the nodal force, FN) of the system (joined and cubic) in the local coordinate system. The values of reference are analytical (see page 6).

Mode II

the joint is open in mode I until the partial damage, then it is requested in mode II. One tests the tangential slopes of stiffness to two values of normal opening. The tangential character incremental of the evolution makes modify the stress tangential during the partial discharge in mode I what returns complex its estimate analytical. Us it testont thus after the discharge by regression.

One notes by:

$$\sigma_{pena}=\sigma_{max}P_{cont}\times(E+L_{cube}\times K_N)/(E+L_{cube}\times K_N\times P_{cont})$$

$$\sigma_t(\delta_n)=K_T\times(1-\delta_n/L_{CT})U_t$$

Quantity tested	Reference	Tolerance ($\%$)
	mode I	
FN, value with the peak	σ_{max} , either $1.D05$	0.10
FN, damaged value	$\beta\sigma_{max}$, or $2.D04$	0.10
FN, value in compression	σ_{pena} , or $-2.D05$	0.10
	mode II	
FT, for two values of δ_n	$\sigma_t(\delta_n)$	0.10

7 the analytical solution for the notations

16 Modelization N

Validation of the joint $3D$ into quadratic with model JOINT_MECA_RUPT

16.1 Characteristics of the modelization

Modelization 3D for the elastic element.
Modelization 3D_JOINT for the element of joint

16.2 Characteristics of the mesh

Many nodes: 32
the elastic element is a HEXA20.
The element of joint is a HEXA20 degenerated (confused nodes).

16.3 Quantities tested and Modes

results I and II

Identical to the modelization Mr.

Mode III

the joint is open in mode I until the partial damage, then it is requested in mode III. One tests the tangential slopes of stiffness to two values of normal opening.

17 Modelization O

Validation of the joint 3D into quadratic with model JOINT_MECA_RUPT

17.1 Characteristics of the modelization

Modelization 3D for the elastic element.
Modelization 3D_JOINT for the element of joint

17.2 Characteristics of the mesh

Many nodes: 24
the elastic element is a PENTA15.
The element of joint is a degenerated PENTA15 (confused nodes).

17.3 Quantities tested and Modes

results I and II

Identical to the modelization Mr. One notes just that $FN=SIGN/2$, because the surface of contact is worth $1/2$.

Mode III

the joint is open in mode I until the partial damage, then it is requested in mode III. One tests the tangential slopes of stiffness to two values of normal opening.

18 Modelization P

Validation of the joint $2D$ into quadratic with model JOINT_MECA_FROT

18.1 Characteristics of the modelization

Modelization in plane strains D_PLAN for the elastic element.
Modelization plane for the element of joint (key word PLAN_JOINT).

18.2 Characteristics of the mesh

Many nodes: The 13
elastic element is a QUAD8.
The element of joint is a degenerated QUAD8 (confused nodes).

18.3 Quantities tested and Mode

results I

One opens the joint until his threshold of strength to traction8voir it[8]. $U=U_{el}$ Then one draws from advantage to arrive at $U=2U_{el}$. It passes in the compression zone $U=-U_{el}/3$. Finally the joint is reloaded up to its point of equilibrium $U=0$. One tests the total response (the resultant of the nodal force, FN) of the system (joined and cubic) in the local coordinate system. The values of reference are obtained analytically (see page 6).

Mode II

the joint is requested in sliding in mode II . One test E then the value of adhesion c . Then one modifies the normal opening in mode I and one makes the sliding return ticket in mode II for $U_t=\pm U_{el}/3$.

Quantity tested	Reference	Tolerance ($\%$)
	mode I	
FN, value with the peak	σ_{max} either 2.D05	0.10
FN, not variation of the value to the peak	σ_{max} or 2.D05	0.10
FN, value in compression	$\sigma_{max}/3$ or -6.66666667D04	0.10
FN, value at the point of equilibrium	0.0	0.10
	mode II	
FT, value of adhesion	C or 1.D05	0.10
FT, value in sliding and adhesion	$-K_N U_t \mu - c$ or -1.2445666D+06	0.10
FT, value in sliding and adhesion	$K_N U_t \mu - c$ or 1.244647D+06	0.10

8 the analytical solution for the notations

19 Modelization R

Validation of the joint $3D$ into quadratic with model JOINT_MECA_FROT

19.1 Characteristics of the modelization

Modelization 3D for the elastic element.
Modelization 3D_JOINT for the element of joint

19.2 Characteristics of the mesh

Many nodes: 32
the elastic element is a HEXA20.
The element of joint is a HEXA20 degenerated (confused nodes).

19.3 Quantities tested and mixed

Mode results I **,** II**,** III
One makes only tests of non regression, by mixing the load patterns. One applies the same loading as in modelization P, but one adds there the loading in the third direction.

20 Modelization S

Validation of the joint $3D$ into quadratic with model JOINT_MECA_FROT

20.1 Characteristics of the modelization

Modelization 3D for the elastic element.
Modelization 3D_JOINT for the element of joint

20.2 Characteristics of the mesh

Many nodes: 24
the elastic element is a PENTA6.
The element of joint is a degenerated PENTA6 (confused nodes).

20.3 Quantities tested and mixed

Mode results I , II , III

One makes only tests of non regression, by mixing the load patterns. One applies the same loading as in modelization P, but one adds there the loading in the third direction.

21 Summary of the results

the numerical results are in agreement with the analytical solution. These tests make it possible to validate the elements of joint, the elements of joint HM in $2\mathrm{D}$ and $3\mathrm{D}$ in the various modes of opening.